

LA-UR

01-4430

Approved for public release;
distribution is unlimited.

Title: CFD ANALYSIS OF A FLOW IN A STORAGE ROOM

Author(s): Snezana Konecni NMI, ESA-DE

Submitted to: Fourth Biennial Tri-Laboratory Engineering Conference on
Modeling and Simulation
DOE Energy Training Center
Albuquerque, New Mexico
October 23-24, 2001



Los Alamos

NATIONAL LABORATORY

Los Alamos National Laboratory, an affirmative action/equal opportunity employer, is operated by the University of California for the U.S. Department of Energy under contract W-7405-ENG-36. By acceptance of this article, the publisher recognizes that the U.S. Government retains a nonexclusive, royalty-free license to publish or reproduce the published form of this contribution, or to allow others to do so, for U.S. Government purposes. Los Alamos National Laboratory requests that the publisher identify this article as work performed under the auspices of the U.S. Department of Energy. Los Alamos National Laboratory strongly supports academic freedom and a researcher's right to publish; as an institution, however, the Laboratory does not endorse the viewpoint of a publication or guarantee its technical correctness.



CFD ANALYSIS OF A FLOW IN A STORAGE ROOM

Snezana Konecni

Los Alamos National Laboratory

Engineering Sciences and Application Division- Design Engineering Group

Los Alamos, New Mexico

ABSTRACT

In this study, a commercial CFD code, CFX5, was used to simulate postulated aerosol release and transport in a ventilated storage room at Los Alamos National Laboratory. A 3-D parametric model of the room was designed in Unigraphics (UG). Details of the room are considered such as lights, wiring conduit, monitors and racks in the model. The ventilation system is modeled since the inlet and outlet ducts are exposed inside the room. A parasolid was exported from UG and imported in the meshing tool of the CFX5 called Build. Boundary conditions were applied according to the flow measurements taken in the room. Inlets were modeled as constant velocity inlet at angle of 25 degrees with respect to the ceiling and outlets as constant velocity as well. Other source of air in the room is the gap under the door that allows air to flow in. The room is kept at negative pressure with withdrawing more air out of the room than what the ventilation supplies. The three-dimensional, steady state flow field was computed using the κ - ϵ turbulence model. The steady-state solution of the flow the room was used for the transient calculation of the spread of released aerosol in the room. The aerosol was simulated assuming dilute, mono-disperse, and neutrally buoyant particles.

Unclassified

Key words: CFD, Flow, Aerosol, turbulence, transient calculations